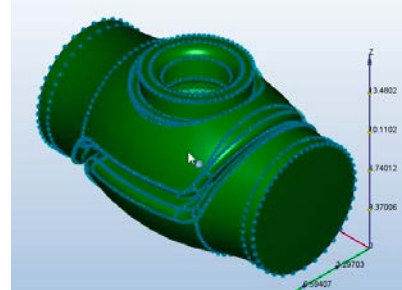


Lesson: Establishing boundary conditions

In this lesson, you are introduced to different types of boundary conditions that can be used to model fluid motion. You will begin by using Fusion 360 to develop a fluid volume that is transferred to the CFD software, and then add the boundary conditions on the fluid.

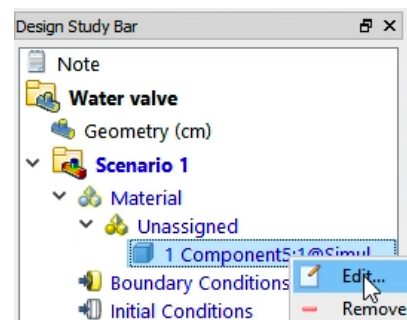


The completed exercise

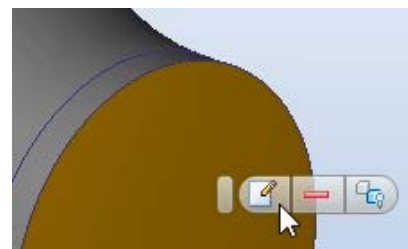
Learning Objectives:

- Understand simulation workflow.
- Set up boundary conditions for a study.
- Apply a mesh to the study model.

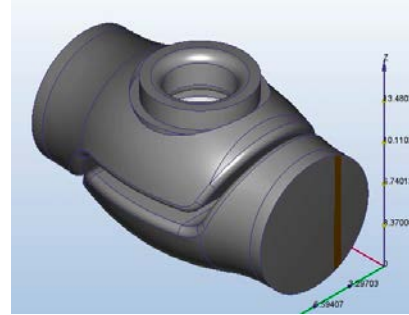
1. Go to the location where you unzipped the Boundary conditions folder and open the file Valve.cfdst. You need to establish the boundary conditions for the fluid as it will pass through the valve. Begin by setting the materials. Right click on Component5 then select Edit. Investigate the options in the Materials control panel; make sure water is selected then click Apply.



2. You now need to set boundary conditions. Click Setup>Boundary Conditions. Select the inlet end of the model then choose Edit from the boundary conditions panel. Change the Units to m/s and the Velocity Magnitude to 1.5 then click Apply.



3. Click Setup>Boundary Conditions. Select the other end of the model then choose Edit from the boundary conditions panel. Change the Type to Pressure, leave the value at 0 and click Apply.



4. Click Setup>Mesh Sizing then click Setup>Autosize. Note that the model is now selected for meshing.

